Study of High Order Discontinuous Galerkin Finite Element CFD Solverf or Aerospace Applications

著者	保江 かな子
号	54
学位授与機関	Tohoku University
学位授与番号	工博第4252号
URL	http://hdl.handle.net/10097/61955

氏　　　　　名	やすえ　かなこ 保江　かな子
授　与　学　位	博士（工学）
学位授与年月日	平成２２年　３月２５日
学位授与の根拠法規	学位規則第４条第１項
研究科，専攻の名称	東北大学大学院工学研究科（博士課程）航空宇宙工学専攻
学位論文題目	Study of High Order Discontinuous Galerkin Finite Element CFD Solver for Aerospace Applications（高次精度不連続ガレルキン有限要素法に基づくCFDソルバーの開発および航空宇宙分野への適用に関する研究）
指　導　教　員	東北大学教授　澤田　恵介
論文審査委員	主査　東北大学教授　澤田　恵介　東北大学教授　中橋　和博 東北大学教授　浅井　圭介

論文内容要旨

Use of Computational Fluid Dynamics (CFD) in practical applications for aerospace engineering seems rapidly expanding in two different areas. In one area, CFD is combined with various optimization methods to look for optimized aerodynamic shapes without resorting to wind tunnel experiments. In the other area, CFD is combined to enhance aerodynamic measurement capability of wind tunnels. In the latter area, CFD is expected to clarify influences of various issues pertinent to wind tunnel measurements and reduce uncertainties in the obtained wind tunnel data. In both application areas, CFD should be capable of handling complicated geometry of real aircraft configurations, and also of resolving various flow features quite accurately. A high order accurate unstructured mesh method is certainly the one that would satisfy these requirements.

Indeed, unstructured mesh methods are very popular in aerospace applications. In these methods, the finite volume formulation is usually chosen because the conservation laws can be rigorously fulfilled for various cell geometries. However, the spatial accuracy of these methods remains usually at most second order. The cause of this lower spatial accuracy can be attributed to the poorly reconstructed dependent variables in computational cell. Conventional reconstruction using the cell-averaged variables in nearby cells tends to loose its accuracy for unstructured mesh particularly when cell geometries are highly skewed.

Recently, the discontinuous Galerkin (DG) finite element method has received attentions because of its ability in achieving higher order spatial accuracy rigorously even on unstructured mesh. In this method, instead of referring to nearby cells as in the finite volume methods, reconstruction of the dependent variables is realized with desired accuracy using the degrees of freedom (DOFs) which are introduced in each cell and evolved in time. Therefore, higher order spatial accuracy can be achieved in the DG method with minimal stencil.

The obvious shortcoming of the DG method is its extremely high computational cost. In order to reduce the computing cost of the DG method, it is certainly necessary to develop an implicit scheme to accelerate the convergence, particularly for

those steady flow problems. In developing implicit schemes, it is very important to have a flexible portability and an easier parallelization capability. These features can be maximized if the implicit scheme is totally cellwise, i.e., no connectivity exists with neighboring cells.

In the first part of this thesis, a cellwise relaxation implicit scheme for unstructured hybrid mesh is proposed as a novel implicit approach for the discontinuous Galerkin CFD solver. The developed CFD solver is successfully validated for various test problems including 3D scalar linear advection problem, inviscid flowfield over ONERA-M6 wing, laminar boundary layer flow over a flat plate and also the turbulent boundary layer flow over a flat plate. A marked stability and fast convergence properties of the developed scheme are indicated for hybrid unstructured meshes.

In order to obtain even faster convergence, the *p*-multigrid scheme is implemented. In addition, the matrix simplification is shown to reduce the computing time without degrading the convergence property of the scheme. Moreover, a high parallel performance of the present scheme is shown for a test problem where the sustained speedup ratio is more than 120 when 128 PEs of SGI Altix 3700 Bx2 are used in the parallel computations. The parallel efficiency is also shown to be very high owing to its cellwise nature.

As an application of the developed CFD solver, the static aeroelasticity problem of a wind tunnel model is chosen. It is well known that aerodynamic characteristics of a wind tunnel model depend critically on Reynolds (Re) number. Therefore, the wind tunnel tests should be conducted with Re number of actual flight condition. Although the National Transonic Facility (NTF) and the European Transonic Wind-tunnel (ETW) have realized such requirement, deformation of wind tunnel models becomes no more negligible in these wind tunnels because a pressurized gas is employed to accomplish the flight Re number. In NTF and ETW, the effect of model deformation can be separated by changing the total pressure and the total temperature simultaneously to obtain a P_0-sweep in which P_0 is varied while both Re number and Mach number are kept constant. In contrast, the high Re number wind tunnel in Japan named as Trisonic Wind Tunnel (TWT) cannot control the total pressure and the total temperature independently. Accurate prediction of aerodynamic performance for high Re number condition in TWT needs at least to isolate the effect of model deformation, and CFD is expected to carry out this role. In the second part of the present thesis, the deformation of a wind tunnel model is formulated as a static aeroelasticity problem and the aerodynamic equilibrium shape of the wind tunnel model is obtained by using the developed high order CFD solver combined with a structural analysis code.

The static aeroelasticity problem of the AGARD-B wind tunnel model installed in supersonic flowfield is first examined. The aerodynamic equilibrium shape is shown to be achieved within several iterations. For the freestream conditions of $M=1.4$, $\alpha=14.9$deg, $Re=4.86\times10^6$, $P_0=167.1$kPa, and $T_0=273$K, the model deformation at the wing tip is shown to become 0.941mm. For this small model deformation, the lift coefficient is decreased 0.002. Similarly, the drag force is decreased by 10 counts. Although these changes in aerodynamic performance are rather modest, it clearly shows that model deformation effects should be accounted for even for a moderate Re number condition.

In addition, a test calculation is attempted to obtain the model deformation for the higher Re number condition in TWT assuming M=1.4, α=8.54deg and P_0=1.4MPa. The corresponding Re number is as large as $40x10^6$. For these freestream conditions, the maximum displacement of about 6mm appears at the wing tip and the lift coefficient decreases by 0.015. It is clearly indicated that aerodynamic performance obtained for the high Re number condition in TWT should be corrected by accounting for the model deformation effect even for the AGARD-B wind tunnel model that has a small delta wing with a small aspect ratio.

Finally, the static aeroelasticity analysis is accomplished for the ONERA-M5 wind tunnel model. For the freestream conditions of M=0.84, α=-1.0deg, Re=$4x10^6$, P_0=220kPa, and T_0=274K, the aerodynamic equilibrium shape is successfully obtained within 3 iterations. The maximum deformation of 3.11mm appears at the wing tip, and the twist of the wind tunnel model gives nose-down. A detailed examination reveals that the deformation is mostly caused by pure bending which reduces the effective angle of attack for the present swept wing.

For this case, the effect of model deformation and also the effect of Re number are evaluated for the freestream condition of M=0.84 and α=-1.0deg. By comparing the computed results for Re=$1x10^6$ and Re=$4x10^6$, it is indicated that an increase in lift coefficient due to Re number effect is totally offset by the effect of model deformation. It is also shown that the amount of drag reduction can be overestimated due to model deformation effect. In addition, a CFD-aided data correction method utilizing the wind tunnel data is discussed.

論文審査結果の要旨

航空宇宙分野における数値流体力学(CFD)には，航空機などの複雑形状の取り扱いが可能な高い形状適合性と，流れ場の物理を正確に捉える高い空間精度が同時に要求される。今日では有限体積法に基づく非構造格子法がこれらの要求を満たす実用コードとして広く用いられるようになった。しかし工学的ツールとしての利用が広まるにつれて，非構造格子上でさらに高い計算精度を実現する新たな手法の開発が求められている。計算セル形状が多岐にわたる非構造格子上でも定式どおりの高い計算精度を達成することが可能な不連続ガレルキン有限要素法が近年注目されているが，計算コストが高く実用コードとして整備されるに至っていない。本研究では，並列計算機上で高い並列効率を容易に実現することが可能なセル緩和型陰的不連続ガレルキン有限要素法を新たに提案して，様々なテスト問題を通して実用性の検証を行うとともに，風洞試験模型の静的空力弾性問題に適用して風洞試験時に生じる模型変形が空力係数に与える影響の詳細な解析を行い，風洞試験データに含まれる模型変形効果の分離や風洞レイノルズ数を変化させた場合のレイノルズ数効果の抽出について議論している。本論文は，これらの研究成果をまとめたものであり，全編5章からなる。

第1章は序論であり，本研究の背景，目的および構成を述べている。

第2章では，不連続ガレルキン有限要素法に基づくCFDソルバーに対する新たな陰的時間積分法として，セル緩和型の陰解法を開発している。セル緩和型は並列計算機との親和性が高く，実際に並列計算機上に実装され高い並列化効率とスケーラブルな性能を得ている。これは，不連続ガレルキン有限要素法に基づく実用CFDソルバーの構築において非常に重要な成果である。

第3章では，セル緩和型陰的不連続ガレルキン有限要素法と構造解析コードを組み合わせて，アスペクト比の小さなデルタ翼を持つAGARD-B風洞検定模型の静的空力弾性問題を解き，空気力と弾性変形が釣り合う空力平衡形状が数度の反復で得られることを示している。また，通常の運用域における風洞気流条件においても模型変形が生じていることや，その変形によって生じる揚力や抵抗などの空力係数の変化が無視できない大きさであることを示している。これは風洞試験データ中に含まれる模型変形効果の分離がCFDの援用によって可能であることを示すものであり，風洞計測技術の高度化において重要な知見である。

第4章では，大きなアスペクト比の主翼を持つONERA-M5風洞検定模型の静的空力弾性問題を解いて空力平衡形状を求め，模型変形が空力係数に与える変化量を明らかにしている。また，レイノルズ数の異なる気流条件に対する解析より，レイノルズ数変化による空力係数の変化と模型変形による空力係数の変化を求め，模型変形効果がレイノルズ数効果で得られる揚力係数の増分を覆い隠すことや，抵抗係数が過大評価されることを示している。これは，従来，経験的に行われてきたレイノルズ変化に伴う空力係数補正がCFDの援用により合理的に行うことが可能であることを示すものであり，航空機空力形状設計技術のより一層の高度化に貢献する重要な成果である。

第5章では，結論を述べている。

以上要するに本論文は，新たに開発されたセル緩和型陰的時間積分法を用いることによって不連続ガレルキン有限要素法が並列計算機上で高い計算効率を達成できること，ならびに風洞試験模型の静的空力弾性問題への適用によって風洞試験データに混在する模型変形効果の分離が可能であり，高レイノルズ数域の風洞試験データの信頼性を高め，さらに航空機の空力形状設計においてこれまで経験的に行われているレイノルズ数補正が合理的に実現できることを示すものであり，航空宇宙工学および数値流体力学の発展に寄与するところが少なくない。

よって，本論文は博士(工学)の学位論文として合格と認める。